AOARD REPORT

2nd Japan-US Symposium on Finite Element Methods in Large Scale Computational Fluid Dynamics

Mar 14-16, 94
T. Davis
AOARD

A summary of the 2nd Japan-US Symposium on Finite Element Methods in Large Scale Computational Fluid Dynamics, conducted Mar. 14-16, 1994 at Chuo University in Tokyo, Japan is presented. Abstracts of all symposium presentations are included. This report is prepared jointly with Dr. Charles Holland, Director, Mathematics & Computer Science Directorate, Air Force Office of Scientific Research (AFOSR/NM) based upon information collected via symposium attendance, review of the symposium proceedings, and a post symposium conversation with Professor Mutsuto Kawahara, Department of Civil Engineering, Chuo University, who served as the Japanese co-chair of the symposium organizing committee.

AIR FORCE OFFICE OF SCIENTIFIC RESEARCH

ASIAN OFFICE OF AEROSPACE RESEARCH AND DEVELOPMENT

TOKYO, JAPAN
UNIT 45002
APO AP 96337-0007
DSN: (315)229-3212
Comm: 81-3-5410-4409

DTIC
ELECTE
APR 0 6 1995
B
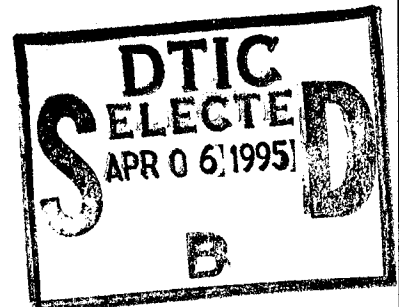
19950321 131

DTIC QUALITY INSPECTED 1

2nd Japan-US Symposium on Finite Element Methods in Large Scale Computational Fluid Dynamics

Abstract

A summary of the 2nd Japan-US Symposium on Finite Element Methods in Large Scale Computational Fluid Dynamics, conducted Mar. 14-16, 1994 at Chuo University in Tokyo, Japan is presented. Abstracts of all symposium presentations are included. This report is based upon information collected via symposium attendance, review of the symposium proceedings, and a post symposium conversation with Professor Mutsuto Kawahara, Department of Civil Engineering, Chuo University, who served as the Japanese co-chair of the symposium organizing committee.

1. Symposium Background, Program and Registration Statistics

The 2nd Japan-US Symposium on Finite Element Methods in Large Scale Computational Fluid Dynamics was conducted Mar. 14-16, 1994 at the Surugadai Memorial Hall of Chuo University in Tokyo, Japan. As the symposium title suggests, the event was the second in a series of biennial joint Japan-US CFD/FEM symposia. The first was held at the University of Minnesota in August, 1992 and the third is tentatively planned for 1-3 April, 1996, also at the University of Minnesota. The symposium series is one of the products of several years of collaborative activity between a group of Japanese researchers centered around the Civil Engineering School at Chuo and the US Army High performance Computing Center located at the University of Minnesota.

The three co-chairmen of the 2nd Japan-US Symposium on Finite Element Methods in Large Scale Computational Fluid Dynamics organizing committee are (including contact information)

Professor Mutsuto Kawahara
Department of Civil Engineering
Faculty of Science and Engineering
Chuo University
1-13-27 Kasuga
Bunkyo-ku
Tokyo, 112 Japan
Tel: (03)3817-1811/1807
Fax: (03) 3817-1803
e-mail: kawa@civil.chuo-u.ac.jp

Professor Tayfun Tezduyar
Professor of Aerospace Engineering and Mechanics
University of Minnesota
Interim Director, Army High Performance Computing Research Center
1100 Washington Ave. South, Suite 101
Minneapolis, MN 55415
Tel: 612-626-8095
Fax: 612-626-1596
e-mail: tezduyar@ahpcrc.umn.edu

1

Professor Thomas J. R. Hughes
Division of Applied Mathematics
Stanford University
Stanford, California 94305-4040
Tel: (415) 723-2040
Fax: (415) 723-1778

The symposium program was divided into a total of twelve sessions, two morning and two afternoon sessions on each of the three days. With the exception of session 4, all were devoted entirely to presentation of papers. Session 4 (Monday afternoon, 14 Mar) consisted of a panel discussion, and is described further in section 2 below. A total of 47 papers was presented, of which 17 were delivered by US based symposium participants. The remaining 30 were presented by Japanese participants. All presentations were delivered in English and all papers and presentation aids were prepared in English.

A total of 143 registrants attended the symposium. Of the indicated total, 23 registrants are US based and the remaining 120 are Japanese.

The symposium proceedings is a collection of extended abstracts of the 47 paper presentations. A post symposium contact with Professor Kawahara indicates that full length versions of all papers are scheduled to be published in special issues of two separate journals: (1) an upcoming special issue of the "International Journal of Numerical Methods in Fluids", John Wiley and Sons, and (2) the premier issue of the new journal "International Journal of Computational Fluid Dynamics", Gordon and Breach Science Publishers. According to Professor Kawahara, publication dates for the special issues are not currently available.

2. Panel Session

Session 4 of the symposium was devoted to a panel discussion. Following introduction of the three invited panelists and a brief description of the panel format, each panelist spoke for about 15-20 minutes, and then entertained questions for a few minutes further. The panel was composed of Dr. Jagdish Chandra, Director, Mathematical and Computer Sciences Division, US Army Research Office, Professor N. Satofuka, Mechanical Systems Engineering, Kyoto Institute of Technology and Dr. Charles J. Holland, Director, Mathematics and Computer Sciences, US Air Force Office of Scientific Research.

Dr. Chandra spoke on the topic "High Performance Computing Issues and Challenges". He identified a list of six major application domains (i.e. Advanced Material Development and Manufacturing, Computational Fluid Dynamics, Biotechnology, Environmental Sciences, Signal and Image Processing and Modeling and Simulation) and then highlighted Parallel and Distributed Computing, Algorithm/Software Development, Graphics and Visualization and Human Resources as the key technical issues in their effective pursuit. Dr. Chandra's concluding remarks summarized a vision to which he referred as "Towards Teraflop Computing".

Professor Satofuka chose "Trends in High Performance Computing in Computational Fluid Dynamics" as his topic. He noted that his background is in finite differences, and he identified Grid Generation, Algorithmic Structures, Computer Resources and Physical Models as the key elements of High

2

Performance Computing in Computational Fluid Dynamics. He devoted an appreciable amount of his discussion to structured and unstructured grid generation techniques.

Dr. Holland spoke on "Critical Issues Facing High Performance Computing". Following is a synopsis of his remarks in the form of quotations from his personal notes. The quoted material is enclosed in brackets.

["I would like to make three points that will impact the use of high performance computing, and which will influence the directions of computer science research at AFOSR.

"(1) We need to think about developing more focused application software capable of use at the departmental level in industry that is flexible (open) enough to incorporate hierarchical modeling and the latest numerical advances. A lot of general purpose finite element codes do not permit enough flexibility to permit correct modeling of key phenomena. Examples include the treatment of fasteners, laminated composites. This limitation restricts the ability to use these packages on real problems. Hierarchical models that permit control of the idealization error are needed as well as methods that allow a priori analysis of the numerical error. We are supporting these ideas at Washington University through the support of B. Szabo.

"(2) Moving beyond simulation, optimal multidisciplinary design is another potential application of high performance computing. In the past, people have attempted to simply graft together their favorite simulation package with a standard optimization package. This often leads to non-convergence and errors in the calculations! I believe that we need to bring together finite element modelers and optimization experts at the beginning to correctly design optimal design packages. We are experiencing some success in this area with a multidisciplinary research team at VPI involving Max Gunzberger and John Burns. They have worked on an "artificial" forebody design for use in wind tunnel simulation at AEDC.

"(3) For progress in computational fluid dynamics, we will need techniques for accurate long time integration of unsteady calculations in order to compute the finite details of shock-vortex calculations and other important time dependent phenomena. There are a range of numerical basics that need to be treated including the correct numerical treatment of time dependent boundary conditions. I would like to mention some recent work of David Gottlieb and colleagues at Brown University. They have developed a technique that resolves the Gibbs' phenomenon which is a problem with discontinuities in spectral methods. In flow calculations, this results in spurious oscillations around shocks. He has shown how to use the spectral information in a post processing scheme to create a new expansion in terms of Gegenbauer polynomials that converges uniformly permitting an accurate capturing of the shock phenomena.]

3. Comments

The English language competence of the Japanese participants in this symposium is very noteworthy. It significantly exceeds the norm for joint Japanese-US conferences. The papers are well written and without exception the English language delivery excellent.

The two central research communities represented at this symposium, the Civil Engineering Department at Chuo University and the Army High Performance Computing Research Center at the University of Minnesota, have developed a very effective working relationship over recent years. Many of the Japanese researchers have visited AHPCRC on sabbaticals, etc. and the U. S. researchers (including the U. S. Army participants) are well aware of who they are and what they do. However, there seems to be little or no participation by the other U. S. military service laboratory organizations.

Although the symposium name indicates an international CFD scope, the participants are more representative of a collection of FEM researchers who have moved from an historical interest in structural mechanics into CFD. The community and the symposium series might benefit from a move to invite participation by representatives from other CFD methods to future events.

The US computer science community is very interested in tracking large scale computing applications in Japan, and the symposium committee responded by scheduling post symposium visits to several Japanese industrial computing facilities. However, with perhaps one notable exception (a visit to Hitachi), very few examples of large scale computing activity were presented. The impression of the participants is that the work exists but was not on display.

4. Abstracts

The symposium proceedings include "extended abstracts" of all 47 presented papers. As noted in section 1 above, the symposium organizers intend to have full length versions of all 47 presentations published, approximately one half in an upcoming special issue of the International Journal of Numerical Methods in Fluids and the remainder in the premier issue of International Journal of Computational Fluid Dynamics. According to the symposium organizing committee, the required arrangements have been made with both journal editorial boards. However, discussions during the course of the symposium indicate that many of the full length papers are still in preparation. Thus, it might well turn out that something less than all full length papers are ever published.

Following is a compilation of selections from the extended abstracts. Typically, depending upon the organization and content of the paper being excerpted, an introductory section (in its entirety) accompanies title and author/organization information. However, in some cases, due to length and or depth of content, the introductory section is abbreviated. Also, in those cases where the excerpted selection cites entries in an accompanying reference list, the corresponding reference list is included.

4

Another Stabilized Q_1Q_1 Incompressible Navier-Stokes Solver Using Explicit Time-Marching*

P.M. GRESHO and S.T. CHAN
Lawrence Livermore National Laboratory, Livermore, CA, USA

ABSTRACT

While on our way to investigating the potential utility of a semi-implicit time-stepping scheme based on a second-order projection method in which the Q_1Q_1 element (bilinear in both velocity and pressure) is stabilized by the simple expediency of replacing the (unstable) matrix representing the Laplacian associated with the consistent discrete projection by a stable but less-consistent matrix in which, for example, spurious pressure modes are traded in for imperfect mass conservation, we have spent some time studying and testing a simpler but related notion: forward Euler time-stepping in which the (same) consistent discrete matrix associated with the pressure Poisson equation (PPE) is again replaced by the same less-consistent matrix. While the final goal is to test the 'approximate' projection method in which a second-order Adams-Bashforth method is applied to the advection terms and trapezoid rule to the remaining terms, and the (approximate) projection matrix problem is solved via the world's best multi-grid scheme, this paper reports on our interim goal of seeing how cost-effective the same matrix replacement would be for the simplest code to write, debug, and understand.

But--as is so often the case with incompressible viscous flow--even forward Euler is not as simple as we had assumed; e.g. the Differential Algebraic Equations (DAE's) associated with the Q_1Q_1 forward Euler scheme are not nearly as 'useful' as one might expect, and the hopes of precluding all pressure modes may not be realized. In fact, we have even learned that the simple and obvious one-for-one matrix replacement starting with the original Index 1 DAE's does *not* yield a method that 'works'--a statement that is easy to support--and we do so below.

Analysis of Shear Layers Based on the Lattice Gas Model

Atsushi YUASA, Takaji INAMURO* and Takeshi ADACHI

Advanced Technology Research Center,
Mitshbishi Heavy Industries, Ltd., 8-1, Sachiura 1-chome, Kanazawa-ku, Yokohama 236, Japan

1. INTRODUCTION

Conventional numerical analyses of fluids are usually performed by using the NavierStokes equation derived from the continuum theory in which fluid is macroscopically regarded as the continuum. As an opposite approach, the "lattice gas model (LGM)" based on a microscopic physical model called "cellular automaton" is recently paid much attention to as a new computational method[1-2]. The LGM imitates motion of particles in a very simplified microscopic world with completely discrete nature. It has begun to be applied to numerical simulations for various complicated physical phenomena[3-5] such as fluid motion, molecular diffusion, chemical reaction and so on.

In this paper, we used the two-color LGM[3-5] and evaluated the validity of this model by applying it to the analysis of shear layers.

* He is currently working with the Department of Chemical Engineering, Kyoto University.

REFERENCES

[1] S. Wolfram, J. Stat. Phys., *45* (1986) 471-526.
[2] U. Frisch, et al., Complex Systems, 1 (1987) 649-707.

[3] D. d'Humieres, and P. Lallemand, Complex Systems, 1 (1987) 599-632
[4] C. Burges, and S. Zaleski, Complex Systems, 1 (1987) 31-50.
[5] D. d'Humieres, P. Lallemand, and G. Searby, Complex Systems, 1 (1987) 633-647.

Finite Element Analysis for Three-Dimensional High Reynolds Number Flows

K. KAKUDA and N. TOSAKA

Department of Mathematical Engineering,
College of Industrial Technology,
Nihon University, Narashino, Chiba 275, Japan

1. INTRODUCTION

It is well known that the conventional Galerkin finite element and centered finite difference approximations lead to sparrows oscillatory solutions for flow problems at high Reynolds numbers. To overcome such oscillations, various upwind schemes have been successfully presented in both frameworks[1].

In our previous work, we presented a finite element scheme based on the PetrovGalerkin weak formulation using exponential test functions for solving the two-dimensional incompressible Navier-Stokes equations at high Reynolds numbers[2]. The method was tested on a square cavity flow and flow around an obstacle, and the numerical results were compared with those of the other existing ones. Consequently, we demonstrated that the method was capable of solving the incompressible NavierStokes equations accurately and in a stable manner for high Reynolds numbers.

The purpose of this paper is to extend the Petrov-Galerkin finite element method using exponential test functions to three-dimensional incompressible viscous flow problems at high Reynolds numbers. In order to show the workability and the validity of its extension to three-dimensional problems, flows in a cubic cavity up to Reynolds number of 10^4 are demonstrated.

REFERENCES

[1] R. Peyret and T.D.Taylor, Computational Methods for Fluid Flow, SpringerVerlag, New York (1983).

[2] K.Kakuda and N.Tosaka, Finite element approach for high Reynolds number flows, Theoretical and Applied Mechanics, 41 (1992), pp.223-232.

[3] A.K.Prasad, C.Y.Perng and J.R. Koseff, Some observations on the influence of longitudinal vortices in a lid-driven cavity flow, AIAA paper, 88-3654-CP, (1988)

Application of FEM/FDM Overlapping Scheme to Heat and Mass Transfer Problem

M. IKEGAWA, M. KAIHO and C. KATO[1]

[1] Mechanical Engineering Research Laboratory, Hitachi, Ltd., Tsuchiura, Ibaraki, Japan

1. INTRODUCTION

There are two major numerical analysis techniques that are commonly applied to computational fluid dynamics: the finite element method (FEM) and the finite difference method (FDM). Finite element method is best suited for accurate simulations of flows within arbitrarily shaped flow geometry, while FDM analysis is best suited for minimizing computational time and storage requirements. To effectively solve flow in a complex region of interest effectively, Nakahashi and Obayashi presented an FDM/FEM

zonal approach to analyze compressible flows in turbine cascades [1]. In that approach, the region near wail boundaries was covered with a boundary-fit grid and the remaining regions were covered with a finite-element mesh. The authors proposed a different technique based on an FEM/FDM overlapping scheme for viscous, incompressible flow, which has been applied to a two dimensional moving boundary problem around high speed trains [2]. In this technique, FEM meshes are used in the region near wall boundaries and FDM meshes are in the other regions. This technique permits accurate simulation of boundary layers and solution of moving boundary problems, with a minimum of computer resources. From a mesh generating point of view, this scheme is thought to be better than conventional schemes, because it is relatively easier to generate an FEM mesh near the complex-shaped wail compared with generating an FEM mesh for the entire calculation domain. In this paper, some numerical results for heat and mass transfer obtained with this calculation scheme are presented.

REFERENCES
[1] Nakahashi, K. and Obayashi, S., FDM-FEM Zonal Approach £ or Viscous Flow Computations Over Multiple Bodies, AIAA paper No. 87-0604, 1987.
[2] Ikegawa, M. et al., FEM/FDM Composite Scheme {or Viscous Incompressible Flow Analysis, to be published in Comput. Methods Appl. Mech. Engrg.
[3] VieceHi, J.A., A Computing Method for Incompressible Flows Bounded by Walls, J. of Computational Physics, Vol. 8, pp. 119-143, 1971.
[4] Ikegawa, M. et al., Three-Dimensional Turbulent Flow Analysis in a Cleanroom by the Finite Element Method, ASME W.A.M., FED 66, pp. 161-167,1988.

Massively Parallel Compressible Flows Computations in Aerospace Applications

S.K. Aliabadi and T.E. Tezduyar

AEM/AHPCRC, Supercomputer Institute, University of Minnesota
1200 Washington Avenue South, Minneapolis, MN 55415, USA

1. INTRODUCTION

We present massively parallel finite element computations of compressible flows with the Connection Machine CM-5 and CM-200. The flow problems we consider typically come from aerospace applications. We use conservation variables formulations of the Euler and Navier-Stokes equations, and to prevent potential numerical oscillations, we use streamline-upwind/Petrov-Galerkin (SUPG) stabilization. For compressible flows, this stabilization technique was first used by Tezduyar and Hughes [1]. In recent years this technique was enhanced by a shock-capturing term [2], and this enhancement rendered this technique very comparable in accuracy to those that followed [1] and formulated in entropy variables [3]. The numerical strategy we use here to solve compressible flow problems with moving boundaries and interfaces--the deformablespatial-domain/stabilized-space-time formulation--was first introduced in the context of incompressible flows [4], and soon after that was applied to compressible flows [5].

Our massively parallel computations of compressible flows started in early 1992, with our first results reported in Tezduyar et al. [6]. A later version of reference [6] appeared as a journal article by Tezduyar et al. [7]. Our parallel implementations are based on the assumption that the mesh is unstructured, and this makes our parallel computations applicable to practical problems with complicated geometries. For time-integration, we typically use implicit methods, although we also have the capability to use--and have in fact used in some problems---explicit methods. Today we are at a point where we have the capability to solve a large class of practical problems, including those involving moving boundaries and interfaces and those in 3D, using implicit formulations with problem sizes over 5,000,000 equations and computational speeds at 12 GigaFlops. We believe this new capability has pushed large-scale finite element computation of compressible flows to a new era in numerical simulation.

ACKNOWLEDGMENTS

REFERENCES

[1] T.E. Tezduyar and T.J.R. Hughes, "Finite Element Formulations for Convection Dominated Flows with Particular Emphasis on the Compressible Euler Equations", AIAA Paper 83-0125.
[2] G.J. Le Beau and T.E. Tezduyar, "Finite Element Computation of Compressible Flows with the SUPG Formulation", *Advances in Finite Element Analysis in Fluid Dynamics* (eds. M.N. Dhaubhadel et al.), FED-Vol. 123, ASME, New York (1991).
[3] F. Shakib, "Finite Element Analysis of the Compressible Euler and Navier-Stokes Equations", Ph.D. Thesis, Mechanical Engineering, Stanford University, 1988.
[4] T.E. Tezduyar, M. Behr and J. Liou, "A New Strategy for Finite Element Computations Involving Moving Boundaries and Interfaces -- The DSD/ST Procedure: 1. The Concept and the Preliminary Numerical Tests", *Computer Methods in Applied Mechanics and Engineering*, 94 (1992) 339-351.
[5] S.K. Aliabadi and T.E. Tezduyar, "Space-Time Finite Element Computation of Compressible Flows Involving Moving Boundaries and Interfaces", *Computer Methods in Applied Mechanics and Engineering*, 107 (1993) 209-223.
[6] T.E. Tezduyar, S. Aliabadi, M. Behr, A. Johnson and S. Mittal, "Massively Parallel Finite Element Computation of Three-dimensional Flow Problems", *Proceedings of the 6th Japan Numerical Fluid Dynamics Symposium*, Tokyo, Japan (1992).
[7] T.E. Tezduyar, S. Aliabadi, M. Behr, A. Johnson and S. Mittal, "Parallel Finite Element Computation of 3D Flows", *IEEE Computer*, October (1993) 27-36.
[8] H. Schlichting, "Bundary-Layer Theory", 7th ed., McGraw-Hill, New York, 1979.

Numerical Investigation of the Nonlinear Galerkin Method
for the Incompressible Navier-Stokes Equations

Takashi NOMURA

University of Tokyo, Tokyo, Japan

1. INTRODUCTION

Temam and his colleagues [1] have been developing the dynamical systems theory for dissipative dynamical systems. It is shown that approximate inertial manifolds, which represent interaction between large and small scale motions, approximate the global attractors of such dynamical systems. In order to investigate their own theory, named nonlinear Galerkin methods, they have carried out computations using a hierarchical finite element for the 2D Burgers equation. We will show a different finite element to implement the nonlinear Galerkin method for the incompressible Navier-Stokes equations.

REFERENCES
[1] Leminie, Pascal & Temam: Computational Mechanics (1993) vol. 11, p.384.
[2] Temam: Infinite-Dimensional Dynamical Systems in Mechanics and Physics, Spfinger-Verlag (1988).
[3] Fotin: Int. J. Numer. Meth. Fluids (1981) vol.1, p.323.
[4] Griffiths: Int. J. Numer. Meth. Fluids (1981) vol. 1, p.347.

Space-Time Finite Element Computation
of Compressible Flow Between Moving Components

G.P. WREN[1], S.E. RAY[2], S.K. ALIABADI[2], and T.E. TEZDUYAR[2]

[1]U. S. Army Research Laboratory, Aberdeen Proving Ground, MD, USA
[2]University of Minnesota, Minneapolis, MN, USA

Recent advances in finite element strategies to solve compressible flow problems have allowed the solution of equations describing challenging applied problems. The problems investigated in this paper involve nearly incompressible, barotropic flow (i.e., flows in which small changes in density lead to large changes in pressure). Such problems challenge the robustness of the numerical method, and, in some cases, limited experimental data with which to compare the solution are available. In general, the physics and dynamics of nearly incompressible flows at high pressure over complicated geometries are not yet fully understood. In particular, better understanding of thermal characteristics and the potential of the flow to remain attached to the boundary and to develop vortices is needed to design injectors for such flows.

The finite element method is ideally suited for the class of problems described previously because of the inherent geometric flexibility of the method. Since grid indexing is not necessary, unstructured grids which are often needed to effectively solve for flow properties over complicated geometries are permitted. In addition, the calculations can be efficiently performed in an element-by-element fashion to carry out the large-scale, time-dependent, nonlinear computations on supercomputers.

In the specific problem examined in this paper, the changes in the geometry of the problem require a formulation that can handle moving boundaries and interfaces. The Deformable-Spatial-Domain/Stabilized-Space-Time (DSD/SST) formulation introduced by the team at the University of Minnesota (UM) in 1990 has this capability. This is an accurate, general-purpose stabilized finite element formulation for computation of unsteady flow problems involving free surfaces, two-liquid interfaces, and fluid-structure and fluid-particle interactions. In the DSD/SST method, the stabilized finite element formulation of the governing equations is written over the space-time domain of the problem, and therefore the deformation of the spatial domain with respect to time is taken into account automatically. With the advanced stabilization techniques used in the DSD/SST formulation, the numerical stability challenges involved in the problem are overcome with minimal numerical dissipation and, therefore, with minimal loss of accuracy.

An arbitary Lagrangian-Eulerian finite element method
for fluid-structure interaction problem

Akira ANJU, Akira MARUOKA and Mutsuto KAWAHARA

Department of Civil Engineering, Chuo University, Tokyo, Japan

1. INTRODUCTION

This paper presents a finite element analysis of a fluid-structure interaction problem, in which the fluid is treated as incompressible viscous flow, and a structure is idealized as a rigid body supported by elastic springs. The arbitrary Lagrangian-Eulerian (ALE) method[1,2] is employed to solve the flow field around the structure, and the fractional step method[2,3] is adopted for the time integration. For the numerical example, the present method is applied to the flow analysis around the oscillating rectangular cylinder.

References

[1] T. Nomura and T.J.R. Hughes, An arbitrary Lagrangian-Eulerian finite element method for interaction of fluid and a rigid body, Comput. Meths. Appl. Mech. Engrg. 95 (1992), pp.115-138.

[2] V. P. Manisamy and M. Kawahara, A factional step arbitrary Lagrangian. n-Eulerian finite element method for free surface density flows, Comp. Fluid. Dyn. 1 (1993), p.57-77.

[3] M. Shimura and M. Kawahara, Two dimensional finite element flow analysis using the velocity correction method, Proc. of JSCE No.398 (1988), p.255-263.

[4] M. Kawahara., H. Hirano, and T. Kodama, Two-step explicit finite element method for high Reynolds number flow passed through oscillating body, Finite Elements in Fluids vol.5 (1984), p.227-261.

Massively Parallel Finite Element Simulation of Incompressible Flows

S. Mittal and T.E. Tezduyar

AEM/AHPCRC, Supercomputer Institute, University of Minnesota
1200 Washington Avenue South, Minneapolis, MN 55415, USA

1. INTRODUCTION

Massively parallel finite element simulations of incompressible flows with the Connection Machine CM-5 and CM-200 are presented. These simulations include flow problems involving moving boundaries and interfaces--such as fluid-body and fluidstructure interactions--and 3D flows. For problems involving moving boundaries and interfaces we use the deformable-spatial-domain/stabilized-space-time formulation which was first introduced by Tezduyar et al [1], and which automatically takes into account the deformation of the spatial domain with respect to time. To update the mesh as the spatial domain deforms with time in this type of problems, special mesh moving techniques as well as automatic mesh moving methods are used to minimize, and in some cases totally eliminate, the need for remeshing (i.e., the need for generating a new set of nodes and elements). Remeshing involves projection errors, and in 3D problems the cost of repetitive mesh generation and parallelization set up (following each remeshing) could be prohibitive. For these reasons, we find mesh update strategies which minimize the frequency of remeshing desirable. If the problem does not involve any moving boundaries or interfaces, then, to avoid the cost associated with space-time formulations, we prefer to use the pressure-stabilizing/Petrov-Galerkin (PSPG) formulation [2], which can be cast in the context of a semi-discrete formulation. The stabilized nature of our formulations allows us to use equal-order interpolation functions for velocity and pressure.

We started implementing our incompressible flow formulations on the Connection Machines in the fourth quarter of 1991. We reported our first results in Tezduyar et al. [3], and we were soon able to solve a large class of practical flow problems, such as fluid-body and simple fluid-structure interaction problems [4] and 3D problems [5-6]. In all these computations the parallel implementation is based on the assumption that the mesh is unstructured, and this gives us a parallel performance with very little dependence on the complexity of the geometry and mesh. Furthermore, all these computations are based on implicit time-integration methods.

These new computational capabilities are now allowing us to solve problems which we were not able to attempt before. Today, essentially all our computations are carried out on the massively parallel platforms of the Connection Machines, with computational speeds which are two orders of magnitude higher than what was available to us two-three years ago on the traditional vector supercomputers. Today we routinely solve timedependent, 3D incompressible flow problems and even 3D problems with moving boundaries and interfaces.

ACKNOWLEDGMENTS

REFERENCES

[1] T.E. Tezduyar, M. Behr and J. Liou, "A New Strategy for Finite Element Computations Involving Moving Boundaries and Interfaces -- The DSD/ST Procedure: I. The Concept and the Preliminary Numericai Tests", *Computer Methods in Applied Mechanics and Engineering,* 94 (1992) 339-351.

[2] T.E. Tezduyar, "Stabilized Finite Element Formulations for Incompressible Flow Computations", *Advances in Applied Mechanics,* 28 (1991) 1-44. [3] T.E. Tezduyar, M. Behr, S. Mittal and A.A. Johnson "Computation of Unsteady Incompressible Flows with the Stabilized Finite Element Methods--Space-Time Formulations, Iterative Strategies and Massively Parallel Implementations", *New Methods in Transient Analysis* (eds. P. Smolinski, W.K. Liu, G. Hulbert and K. Tamma), AMD-Vol. 143, ASME, New York (1992) 7-24.

[4] S. Mittal and T.E. Tezduyar, "Massively Parallel Finite Element Computation of Incompressible Flows Involving Fluid-Body Interactions", to appear in *Computer Methods in Applied Mechanics and Engineering.*

[5] T.E. Tezduyar, S. Aliabadi, M. Behr, A. Johnson and S. Mittal, "Massively Parallel Finite Element Computation of Three-dimensional Flow Problems", *Proceedings of the 6th Japan Numerical Fluid Dynamics Symposium,* Tokyo, Japan (1992).

[6] T.E. Tezduyar, S. Aliabadi, M. Behr, A. Johnson and S. Mittal, "Parallel Finite Element Computation of 3D Flows", *IEEE Computer,* October (1993) 27-36.

PARALLEL FINITE ELEMENT STRATEGIES FOR LARGE SCALE COMPUTATION OF SHALLOW WATER FLOWS AND CONTAMINANT TRANSPORT

Kazuo Kashiyama[1], Hanae Ito[1], Marek Behr[2] and Tayfun Tezduyar[2]

[1] Chuo University, Tokyo, JAPAN
[2] University of Minnesota, Minneapolis, MN, USA

1. INTRODUCTION

Finite element computations of shallow water flows and contaminant transport can be applied to many practical problems: design of river, coastal and offshore structures, disaster prediction and other applications related to hydrodynamic, thermal and chemical transport behavior in oceans, lakes, and rivers. In this context, the finite element method is applicable to complicated water and land configurations. In practical computation of this type of problems, it is essential to use methods which are as efficient and fast as the available hardware allows. Also, in this type of problems, computations need to be carried out over long time durations to properly simulate and predict the phenomena of interest.

In recent years, massively parallel finite element computations have been successfully applied to several large-scale compressible and incompressible flow problems, including those involving moving boundaries and interfaces and those in 3D [1]. These computations demonstrated the availability of a new level of finite element computational capability to solve practical flow problems. With the need for a high-performance computing environment to carry out simulations for practical problems in shallow water flows and contaminant transport, in this paper we present and employ a parallel explicit finite element method for computations based on unstructured grids. The finite element discretizations are based on a three-step explicit formulation both for the shallow water equations and the advection-diffusion equation governing the contaminant transport. In these discretizations, for numerical stabilization, we use selective lumping [2] for the shallow water equations and the streamline-upwind/Petrov-Galerkin (SUPG) technique [3] for the advectiondiffusion equation. Parallel implementation of these unstructured grid-based formulations

carried out on the Connection Machine CM-5. As an example, we carry out simulation of the effect of tidal waves on the Tokyo Bay and the spread of a pollutant injected into the Tokyo Bay.

ACKNOWLEDGMENT

11

REFERENCE

[1] T.E. Tezduyar, M. Behr, S. Mittal and A.A. Johnson, Computation of unsteady incompressible flows with the stabilized finite element methods - Space-time formulations, iterative strategies and massively parallel implementations, New Methods in Transient Analysis, in P. Smolinski, W.K. Liu, G. Hulbert and K. Tamma (eds.), AMD-Vol. 143, ASME, New York. 7-24. (1992)

[2] M. Kawahara, H. Hirano, K. Tsubota and K. Inagaki, Selective lumping finite element · method for shallow water flow, Int. J. Numer. Methods Fluids, Vol.2 (1982), 89-112.

[3] T.E. Tezduyar and T.J.R. Hughes, Finite element formulations for convection dominated flows with particular emphasis on the Euler equations, AIAA paper 83-0125, Proceedings of AIAA 2 I st Aerospace Science Meeting, Reno, Nevada. (1983).

[4] C.B.Jiang, M. Kawahara, K. Hatanaka and K. Kashiyama, A three-step finite element method for convection dominated incompressible flow, Comp. Fluid Dyn. Journal, Vol. 1 (1993), 443-462.

[5] K. Kashiyama, H. Ito, M. Behr and T.E. Tezduyar, Three step explicit finite element computation of tidal flow involving moving boundary on Connection machine, (in preparation)

[6] M. Behr and T.E. Tezduyar, Finite element solution strategies for large-scale flow simulations, Comp. Meth. Appl. Mech. Eng., (in press).

[7] Z. Johan, K.K. Mathur, K.K., S.L. Johnsson and T.J.R. Hughes, An efficient communication strategy for finite element methods on the connection machine CM-5 system, Thinking Machine Technical Report, Cambridge, MA, 1993 and Comp. Meth. Appl. Mech. Eng. (in press)

[8] K. Kashiyama and T. Okada, Automatic mesh generation method for shallow water flow analysis, Int. J. Numer. Methods Fluids, Vol. 15, 1037-1057. (1992)

A Strategy for Implementing Implicit Finite Element Methods for Incompressible Fluids on the CM-5

J.G. KENNEDY[1], V. KALRO[2], M. BEHR[2], and T.E. TEZDUYAR[2]

1 Thinking Machines Corporation
    245 First Street, Cambridge, MA 02142, USA
2   AEM/AHPCRC, Supercomputer Institute, University of Minnesota
    1200 Washington Avenue South, Mineapolis, MN 55415, USA

1. INTRODUCTION

A parallel implementation of an implicit finite element formulation for incompressible fluids on a distributed-memory massively parallel computer is presented. Data structures as well as data decomposition and data communication algorithms designed for distributed-memory computers are discussed in the context of high level language constructs from High Performance Fortran. The discussion relies primarily on abstract features of the hardware and software environment and should be applicable, in principle, to a variety of distributed-memory systems. The implementation is carried out on a Connection Machine CM-5 system with high performance communication functions.

ACKNOWLEDGMENT

REFERENCES

[1] T.E. Tezduyar, S. Mittal, S.E. Ray and R. Shih, "Incompressible flow computations with stabilized bilinear and linear equal-order-interpolation velocity-pressure elements," *Computer Methods in Applied Mechanics and Engineering*, vol. 95 (1992), 221-242.

[2] M. Behr, L.P. Franca and T.E. Tezduyar, "Stabilized finite element methods for the velocity-pressure-stress formulation of incompressible flows," *Computer Meth. in Applied Mechanics and Engineering*, vol. 104 (1993), 31-48.

[3] T.E. Tezduyar, M. Behr and J. Liou, "A new strategy for finite element computations involving moving boundary and interfaces - the deforming-spatialdomain/space-time procedure: 1. The concept and the preliminary tests," *Computer Methods in Applied Mechanics and Engineering*, vol. 94 (1992), 339-351.

[4] Z. Jobart, K.K. Matbur, S.L. Johnsson and T.J.R. Hughes, "An efficient communication strategy for finite element methods on the Connection Machine CM-5 system," *Computer Methods in Applied Mechanics and Engineering* (to appear).

[5] A. Pothen, H.D. Simon and K.P. Liou, "Partitioning sparse matrices with eigenvectors of graphs," *SIAM Journal of Matrix Analysis and Applications*, vol. (1990), 430-452.

THREE-DIMENSIONAL SLOSHING ANALYSIS BY ARBITRARY LAGRANGIAN-EULERIAN FINITE ELEMENT METHOD

Takashi OKAMOTO[1] and Mutsuto KAWAHARA[2]

[1] NKK Corporation, Kanagawa, JAPAN
[2] Chuo University, Tokyo, JAPAN

1. INTRODUCTION

Sloshing is a free surface flow problem which occurs in a tank subjected to forced oscillation. The main difficulty of computation in large amplitude sloshing analysis arises from the fact that both the calculations of an unknown free surface position and the free surface condition should be satisfied simultaneously, and in addition the fluid is strongly distorted. To overcome this point, a new Arbitrary Lagrangian-Eulerian finite element method for three-dimensional free surface flow problems in a tank was developed.

REFERENCE

[1] T.Okamoto and M.Kawahara: Two-Dimensional Sloshing Analysis by the Arbitrary Lagrangian-Eulerian Finite Element Method", Proceedings of Japan *Society* of Civil Engineers, No.441/1-18. pp.39-48, January 1992.

POTPOURRI OF RECENT DEVELOPMENTS IN STABILIZED METHODS FOR CFD

Thomas J.R. Hughes

Professor
Division of Applied Mechanics Department of Mechanical Engineering'
Stanford University
Stanford, California 94305
U.S.A. 415/723-2040 415/723-1778 FAX

Abstract

Stabilized finite element methods such as SUPG and Galerkin/least squares (GLS) have proved to be effective technologies for solving the Navier-Stokes equations and general advective-diffusive systems. The methods were initially developed for incompressible flows in

[1] and later extended to compressible flows [25]. Our work in compressible flows has emphasized the use of entropy variables. Although the method based on entropy variables possesses unique properties, it is very appealing to extend the formulation to other sets of variables, such as primitive variables. Hughes and Tezduyar [6], Hansbo and Johnson [7] and Aliabadi, Ray and Tezduyar [8] have already developed similar methods within the context of conservation variables. Possible benefits of the availability of the various formulations include easier implementation of the GLS finite element method within existing codes and extension of the range of turbulence models that can be used. The use of primitive variables or entropy variables also allows the solution of incompressible and compressible flows within the same program.

The success of stabilized finite element methods for the laminar Navier-Stokes equations has motivated extension to turbulent flows. We consider Reynoldsaveraged systems which include a turbulent kinetic energy transport equation ("oneequation models"). We identify a turbulent entropy production (in)equality associated with such systems and show that it may be automatically embedded within our numerical formulation, thus incorporating the fundamental nonlinear stability condition *ab initio.*

We also present a few numerical calculations illustrating the preceding ideas. In addition, we present applications on massively parallel computers. This area is one of the most exciting in the history of scientific computation. The engineering potential for such machines is truly mind boggling. We briefly illustrate issues of load balancing via a parallelized domain decomposition algorithm, and present timings for large meshes on a contemporary platform, the Connection Machine CM-5.

<u>References</u>

1. Brooks, A.N. and Hughes, T.J.R.   Streamline Upwind Petrov-Galerkin Formulations for Convection Dominated Flows with Particular Emphasis on the Incompressible Navier-Stokes Equations, *Comput. Methods Appl. Mech. Eng.,* 32, 199-259 (1982).

2. Hughes, T.J.R., Franca, L.P. and Mallet, M. - A New Finite Element Formulation for Computational Fluid Dynamics: I. Symmetric Forms of the Compressible Euler and Navier-Stokes Equations and the Second Law of Thermodynamics, *Comp. Methods Appl. Mech. Eng.,* <u>54,</u> 223-234 (1986).

3. Hughes, T.J.R. and Mallet, M. - A New Finite Element Formulation for Computational Fluid Dynamics: III. The Generalized Streamline Operator for Multidimensional Advective-Diffusive Systems, *Comput. Methods Appl. Mech. Eng.,* <u>58,</u> 305-328 (1986).

4. Hughes, T.J.R. and Mallet, M. - A New Finite Element Formulation for Computational Fluid Dynamics: IV. A Discontinuity-Capturing Operator for Multidimensional Advective-Diffusive Systems, *Comput. Methods Appl. Mech. Eng.,* <u>58,</u> 329-336 (1986).

5. Hughes, T.J.R., Franca, L.P. and Hulbert, G. - A New Finite Element Formulation for Computational Fluid Dynamics: VIII. The Galerkin/least-squares method for Advective-Diffusive Equations, *Comput. Methods Appl. Mech. Eng.,* <u>73,</u> 173-189 (1989).

6. Hughes, T.J.R. and Tezduyar, T.E. - Finite Element Methods for First-order Hyperbolic Systems with Particular Emphasis on the Compressible Euler Equations, *Comput. Methods Appl. Mech. Eng.,* 45, 217-284 (1984).

7. Hansbo, P. and Johnson, C. - Adaptive Streamline Diffusion Methods for Compressible Flow using Conservation Variables, *Comput. Methods Appl. Mech. Eng.,* <u>87,</u> 267-280 (1991).

8.   Aliabadi, S.K., Ray, S.E. and Tezduyar, T.E. - SUPG Finite Element Computation of Viscous Compressible Flows Based on the Conservation and Entropy Variables Formulation, University of Minnesota Supercomputer Institute Research Report UMSI 92/136 (1992).

New Theory of Hybrid-Upwind Technique and Discrete Del Operator for Finite-Element Method

Takahiko TANAHASHI and Yoshiatsu OKI

* Faculty of Science and Technology, Keio University, 3-14-1, Hiyoshi, Kohoku-ku, Yokohama 223, Japan

1 .Introduction

The most important problem of computational fluid dynamics is the accurate prediction of turbulent phenomena of advection-dominated flows at high Reynolds numbers. In order to accomplish this prediction, high resolution schemes have been developed in each field of finite difference, finite volume and finite element methods, for examples, the third-order upwind method and QUICK by Leonard (1979) in the finite difference field, MUSCL by Van Leer (1979) and SIMPLE by Patanker (1980) in the volume field, SUPG by Hughes et al (1977), BTD by Dukowicz-Ramshow (1979), GSMAC by Tanahashi et al.(1990), upstream-downstream selective elements method by Tabata-Fujita (1991) and exponential-type Petrov-Galerkin finite element method by Kakuda-Tosaka (1992). However, upwind technology in the finite element field compared with the other numerical fields has not been advanced. The finite analytic method by Chen (1987) in the finite difference field has gotten attractive. In this method, local exact solution of steady advection- diffusion equation is used in order to simplify algorithms and to obtain the stability at high Reynolds numbers.

The purpose of this paper is to extend the finite analytic method in the finite difference field to the finite element field. This possibility is attained to use the dual spaces composed of adjoint operators. Accurate interpolation function and upwind-weighting function are made at the same time in each dual spaces. Here we introduce a discrete del operator and an adjoint discrete del operator in the dual spaces. Those operators reduce the capacity of diffusion matrix and get the high speed calculation in the finite element method. The basic idea of stability is the element average of continuity equation and the node average of momentum equation. This idea in the unstructured mesh is a generalization of the staggered mesh in the MAC method. That is to say, the conservation of mass and momentum is done in the different domain. After mixing the two equations in the difference domain, we will get the numerical solution of the incompressible Navier- Stokes equation. In this paper, a fundamental theory of discretization for finite element methods will be discussed.

References

(1) Leonard, B.P., *Comp. Method in AppL Mech, and Eng.*   19(1979), *59.*

(2) Vain Leer, B., *J. Comp. Shys.*   32(1979), 101.

(3) Patamker, S.V., *Numerical Heat Transfer and Fluid Flow,*   (1980), McGraw-Hill.

(4)   Hughes, T.J.R. et al., *Comp. Method in AppL Mech. and Eng.*   54(1986), 341.

(5)   Dukowicz, J. and Ramshow, J., *Z Comp. Phys.*   13(1979), 32.

(6)   Tama/tashi, T., et al., *Inter. J. Nume. Meth. in Fluid*   11(1990), *479.*

(7)   Taburn, M. and Fujiran, *S., J. Comp. and Appl. Mech.*   38(1991), 411.

(8)   Kakuda, K. and Tosaka, N., *Theory. and AppL Mech:*   42(1992), 223.

(9)   Chen, C.J., *Handbook of Numerical Heat Transfer Chap. 17,*   (1987), John-Wiley.

Third-Order Upwind Finite Element Simulation of Flow around Twin Square Cylinders

Norio Kondo, Toshio Nishimura and Seiji Yamada

Nihon University, Chiba, Japan

1. INTRODUCTION

In this paper numerical results of flow around twin square cylinders are presented on the basis of numerical simulation of the incompressible Navier-Stokes equations using a third-order upwind finite element scheme[1]. It is well known that a flow pattern around the twin square cylinders is more complicated than a flow pattern around a square cylinder because of interference between the Karman vortices behind the twin square cylinders. We therefore show profiles of the interference of the Karman vortices occurred behind the twin square cylinders.

Some high performance techniques of the upwind finite element scheme based on the choice of up- and downwind points

SHOICHI FUJIMA and MASAHISA TABATA

University of Electro-Communications, Chofu, Tokyo, Japan

1. INTRODUCTION

In [1] we have developed an upwind finite element scheme for the incompressible Navier-Stokes equations at high Reynolds numbers. The idea of the upwind technique is based on the choice of up- and down-wind points (see Fig.1). We refer to [2,3] for the details and the advantages of this scheme. Here we only state that the scheme has a potential to approximate the convection term in third-order accuracy. Thus, it can capture the behavior of high Reynolds number flows well.

In this paper we present some techniques for high performance implementation of the scheme in terms of computation speed and memory size.

REFERENCES

1. M. Tabata and S. Fujima, An upwind finite element scheme for high-Reynoldsnumber flows, Int. J. Numer. Meths. Fluids, 12(1991), 305-322.

2. M. Tabata and S. Fujima, Finite-element analysis of high-Reynolds number flows past a circular cylinder, J. Comput. Appl. Math., 38(1991), 411-424.

3. S. Fujima, M. Tabata and Y. Fukasawa, Extension to three dimensional problems of the upwind finite element scheme based on the choice of up- and down-wind points, to appear in Comput. Meth. Appl. Mech. £ Engrg.

4. M. Bercovier and O. Pironneau, Error estimates for finite element method solution of the Stokes problem in the primitive variables, Numer. Math., 33(1979), 211-224.

5. N. Kikuchi et M., Recent development of finite element methods in the highspeed computing environment, in: Super-Computing, --Applications, Algorithms and Architectures, J. Kondo (ed.), 20-71, Springer-Verlag, Tokyo, 1991.

Data Parallel Finite Element Techniques for
Compressible Flow Problems

Z. JOHAN[1], K.K. MATHUR[1], S.L. JOHNSSON[1] and T.J.R. HUGHES[2]

[1] Thinking Machines Corporation, Cambridge, MA, USA
[2] Division of Applied Mechanics, Stanford University, CA, USA

1. INTRODUCTION

We present a brief description of a finite element Solver implemented on the Connection Machine CM-5 system. A more detailed presentation of the issues in such an implementation can be found in [1, 2].

REFERENCES

[1] Z. Johan, T.J.R. Hughes, K.K. Matbur and S.L. Johnsson, A data parallel finite element method for computational fluid dynamics on the Connection Machine system, Computer Methods in Applied Mechanics and Engineering, vol. 99 (1992) pp. 113-134.

[2] Z. Johan, K.K. Mathur, S.L. Johnsson and T.J.R. Hughes, An efficient communication strategy for finite element methods on the Connection Machine CM-5 system, Thinking Machines Technical Report No. 256 (1993).

[3] A. Pothen, H.D. Simon and K.-P. Liou, Partitioning sparse matrices with eigenvectors of graphs, SIAM Journal of Matrix Analysis and Applications, vol. 11 (1990) pp. 430-452.

Element-by-Element Solution Technique for
the Consistent Pressure Equation

Akira MIZUKAMI

NK-EXA Corporation, 1-8-27, Konan, Minato-ku, Tokyo, Japan

1. INTRODUCTION

In this paper, we present a finite element solution procedure for solving large scale time-dependent incompressible viscous flow problems. It is a velocity-explicit/pressure-implicit time-marching algorithm. For memory-saving, the Q_1-P_0 elements are employed and the discrete pressure Poisson equation is consistently derived from the mixed finite element formulation. The matrix system is solved by an element-byelement conjugate gradient method, which is cost-effective in both storage and CPU time.

Parallel Computation of Unsteady Flows
on Network of Workstations

H.U. AKAY and A. ECER

Department of Mechanical Engineering
Indiana University-Purdue University at Indianapolis Indianapolis, Indiana 46202, USA

1. INTRODUCTION

Parallelization of a flow solver using a machine independent data base program GPAR [1], developed primarily for parallelization of CFD programs, is presented. In this approach the solution block

domain is subdivided into a number of solution blocks greater than or equal to the number of available computers. The blocks are distributed to available computers on a network to solve the same problem in parallel. The data base program handles blocks and their interface information. The machine independence is achieved by utilizing a machine portable parallel library APPL [2] developed at NASA Lewis Research Center. The flow solver consists of separate block and interface solvers [3]. A given computational grid is subdivided into blocks using an automatic divider algorithm which prepares the block and interface information to GPAR. An explicit time integration scheme is used following an upwinded finite element formulation of unsteady Euler equations.

Since for accurate calculation of lift and pressure coefficients around aerodynamic configurations large size problems must be solved for many time steps, memory and CPU requirements of unsteady flow solvers are rather excessive. The parallel approach presented herein provides a practical utilization of networked workstations for this purpose.

REFERENCES

[1] H.U. Akay, R.A. Blech, A. Ecer, D. Ercoskun, W.B. Kemle, A. Quealy, and A. Williams, "A Database Management System for Parallel Processing of CFD Algorithms," *Parallel Computational Fluid Dynamics '92,* Ed. R.B. Pelz, et al, North-Holland, Amsterdam, 1993, pp. 9-23.
[2] A. Quealy, G.L. Cole, and R.A. Blech, "Portable Programming on Parallel/Networked Computers Using the Application Portable -Parallel Library (APPL)," *NASA Technical Memorandum 106238,* July 1993.
[3] A. Ecer, H.U. Akay, W.B. Kemle, H. Wang, D. Ercoskun, and E. Hall, "Parallel Computation of Fluid Dynamics Problems," *Computer Methods in Applied Mechanics and Engineering,* 1994, (in print).

PARALLEL PROCESSING OF
THE CELLULAR AUTOMATON METHOD

Yukihiko INOUE[1], Kazuyuki KATSURAGI[2], Osamu UKAI[1], and Takeshi ADACHI[1]

Mitsubishi Heavy Industries, Ltd.
[1] Advanced Technology Research Center, Yokohama, Japan
[2] Engineering Systems Center, Kobe, Japan

1. INTRODUCTION

The cellular automaton method (the CA method) is a particle simulation method for an incompressible fluid [1][2]. Recently, some complicated flows with multi components have been simulated through the microscopic approach of the CA method (e.g. immiscible two phase flow [3], share layer analysis [4]).

On the other hand, it is known that the discrete structure of the CA method is suited for parallel processing. In fact, we have developed the parallel CA method with 1-dimensional domain decomposition technique and have obtained the speedup factor of more than 63 by a Fujitsu AP1OO0 with 64 processors in 327680 node calculation [5]. The high efficiency of the parallel CA method will be maintained in massively parallel processing (MPP) for the practical use.

In this paper, Poiseuille flow and immiscible two phase flow simulation by the parallel CA method are presented to estimate the suitability for the MPP.

REFERENCES

[1] U.Frisch, D.d'Humieres, B.Hasslacher, P. Lallemand, Y.Pomeau and J.Rivet, Complex Systems 1 (1987) 649-707.
[2] S.Wolfram, J.Stat.Phys. 45 3/4 (1986) 471-526.
[3] D.Rothman and J.Keller, J.Stat. Phys. 52 3/4 (1988) 1119-1127.
[4] A.Yuasa, T.Inamuro, and T.Adachi, 5th ISCFD proceedings (1993).
[5] Y.Inoue, K.Katsur~gi, and O.Ukai, 5th ISCFD proceedings (1993).

Computer Aided Analysis of Convective Instability
in a Bounded Cylindrical Fluid Layer

Robert L. SANI[1] and Damien VEYRET[2]

[1] University of Colorado, Boulder, CO USA
[2] I.U.S.T.I., Universite de Provence, Marseille, France

The analyses of buoyancy-driven flows in fluid layers heated from below has been the of many theoretical and experimental investigations since the observations of Tyndall in 1863 [1] and the analysis of Lord Rayleigh in 1916 [2]. Such fundamental aspects as the onset of convective instability, transition to periodic and non-periodic motions, influence of the form of the perturbation and boundary conditions, etc. have been extensively considered via linear and weakly nonlinear analyses. More recently attention has again been focused on the effect of rigid lateral boundaries on the threshold of stability and the form of the ensuing flow; in particular, rectangular and circular cylindrical domains have been investigated over a range of aspect ratio. Aside from the obvious fundamental aspects of such studies, insight is also gleaned into possible convection in the vapor, or liquid, nutrient phase in certain crystal-growing techniques as, for example, the Bridgman technique (Carruthers [3]). The interest here is primarily in circular cylindrical domains in both a 1-g terrestrial as well as a low-g near-earth orbit where the residual gravity vector can vary both in magnitude and direction as, for example, in the g-jitter phenomena. (See, for example, Ostrach [4]).

A number of studies devoted to convection in vertical confined cylinders heated from below have appeared in the literature but only a few have dealt with the rigid (no-slip) wall case and only a very limited number of fully three-dimensional numerical simulations are available. (See, for example, Smutek et al. [5] for gases, Pr=0.7 and Neumann [6] for liquids, Pr=6.7). The previous theoretical studies were limited to linear and weakly nonlinear stability analyses. (See Hardin et al. [7] and Hardin and Sani [8] for a summary and new results.) While these analyses establish that the onset and slightly supercritical state of steady convection is strongly dependent on aspect ratio (Charlson and Sani [9] and Buell and Catton [10]), they are silent about the flow and structure at elevated supercritical states. The current study is focused on using transient, 3D numerical simulations to confirm previous results as well as to explore the supercritical flow and structure in such systems.

[1] V. Volkovisky, Publ. Sci. Tech. du Minist. de l'Aer, #151 (1939).

t2] Lord Rayleigh, Phil. Mag. 32, 529 (1916). [3] J.R. Carruthers, Thermal convection instabilities relevant to crystal growth from liquids. In Preparation and Properties of Solid State Materials (ed. W.R. Wilcox & R.A. Lefever), vol. 3, pp. 1-121. Dekker. (1977).

[4] S. Ostrach, Low-gravity fluid flows, Ann. Rev. Fluid Mech. 14, pp. 3 13-345. (1982)

[5] C. Smutek, B. Roux, P. Bontoux, & G. DeVahl Davis, 3D finite differences for natural convection in cylinders. In Notes on Numerical Fluid Mechanics (ed. M. Pandolt & R. Piva), vol. 7, pp. 338-345. Vieweg (1983).

[6] G. Neumann, Berechnung der thermischen, Auftriebskonvektion in Modellsystemen zur Kristallzuchtung. Doktor-Ingenieur Dissertation Der Technischen Facultat der Universitat Erlangen-Nurnberg, Germany. (1986).

[7] G.R. Hardin, R.L. Sani, D. Henry and B. Roux, Buoyancy-driven instability in a vertical cylinder: binary fluids with Soret effect. Part 1: General theory and stationary stability results, Int'l. J. Numer. Methods Fluids, 10, 79-117 (1990).

[8] G.R. Hardin and R.L. Sani, Buoyancy-driven instability in a vertical cylinder: Binary fluids with Soret effect. Part II: weakly non-linear solutions, Int'l. J. Numer. Methods Huids, 17, 755-786 (1993).

[9] G.S. Charlson & R.L. Sani, On thermoconvective instability in a bounded cylindrical fluid layer. Int'l. J. Heat Mass Transfer, 14, 2157-2160. (1971).

[10] J.C. Buell & I. Catton, The effect of wall conduction on the stability of a fluid in a right circular cylinder heated from below. J. Heat Transfer 105, 255-260. ASME (1983).

[11] P.M. Gresho and S.T. Chart, On the theory of semi-implicit projection methods for viscous incompressible flow and its implementation via a finite element method that also introduces a nearly-consistent mass matrix, Part 2: Implementation, Int. J. Num. Meth. Fluids, 11, 621-660. (1990).

[12] H. Daniels, PASTIS-3D, UCRL-MA-111833, Lawrence Livermore National Laboratory (1992).

FINITE ELEMENT ANALYSIS ON TEMPERATURE AND FLOW FIELD IN A LARGE SCALE ATRIUM

T. Saito, S.Higuchi, S. Ohgaki, Y. Ozeki and Y. Sonda 1

Research Center, Asahi Glass Co., LTD., Yokohama, Japan

INTRODUCTION

For effective control of the indoor climate in an atrium, it is very important to predict the and flow fields during design. One of the methods for this prediction is numerical simulation. We have developed the numerical simulation system, which consists of three subsystems as follows; (1) simulation system. of the direct solar radiation, the diffused solar radiation and the diffused reflection of the solar radiation, (2) simulation system of the radiative heat transfer, (3) simulation system of the temperature and flow distribution based on k - e model of turbulence solved by GSMAC3D-FEM[1,2].

On the other hand, to verify the accuracy and the reliability of those systems, comparison between results from experiments and numerical analysis was carried out in the test atrium for experiments. As a result of comparison, it became obvious that results of numerical analysis agreed with those of experiments by coupling the calculation of radiative heat transfer and that of convective heat transport.

ACKNOWLEDGMENTS

REFERENCES

[1IT Saito . 1989. GSMAC3D(II) Numerical Simulation of Heat Transfer and Fluid Flow, Reports of the Research Laboratory, Asahi Glass Co.,Ltd., Vol39, No. 1, 1989, pp, 1-36. [2IT Saito. 1991. Numerical Simulation of Turbulent Flow by Using the GSMAC3D Method (partl), Reports of the Research Laboratory, Asahi Glass Co.,Ltd., Vol41, No.2, 1991, pp. 111-158.[3]T Saito. Analysis on Temperature and Flow Field in the Atrium (partl-parts), Summaries of Technical Papers of Annual Meetings, Architectural Institute of Japan,D,1991,pp.521-530.[4]Y Ozeki .Analysis on Temperature and Flow Field in the Atrium (partl-part2), Room Air Convection and Ventilation Effectiveness, ASHRAE, pp.479-500. [5]Y Ozeki, Analysis on Temperature and Flow Field in the Atrium (part6-part9), Summaries of Technical Papers of Annual Meetings , Architectural Institute of Japan, D, 1992, pp551-558.

Massively Parallel Finite Element Analysis of Large-Scale Crystal Growth Processes: Rotating and Coupled Flows

Q. XIAO, A.G. SALINGER, Y. ZHOU, and J.J. DERBY

University of Minnesota, Minneapolis, MN 55455-0132, USA

1. INTRODUCTION

State-of-the-art, high-performance computing has proven to be an invaluable tool for addressing a spectrum of issues in manufacturing technology. Understanding the complex physical phenomena involved with manufacturing processes poses challenges comparable to the "Grand Challenge" supercomputer problems of the pure physical sciences and is, perhaps, even more compelling due to the obvious economic import of manufacturing activity.

Our interests center on processes employed to produce advanced materials, such as large, single crystals for photonic and electronic devices [1 3] and *ceramic* materials [4,5]. Such processes are characterized by complex geometries which are often time-dependent, coupled incompressible flows, heat

20

and mass transfer, chemical reaction, and phase change. As the technological complexity of such processes continues to grow, increasingly powerful techniques must be employed for solution of the mathematical models which describe their physical behavior [6].

Here, we report on continuing progress in developing finite element methods based on algorithms which perform effectively on distributed memory, massively parallel supercomputers, such as the Thinking Machine Corporation CM-5. Our initial efforts were reported in [7].

REFERENCES

[1] J.J. Derby, L.J. Atherton, and P.M. Gresho, An integrated process model for the growth of oxide crystals by the Czochralski method, J. Crystal Growth, vol. 97 (1989), pp. 792-826.
[2] S. Brandon and J.J. Derby, Heat transfer in vertical Bridgman growth of oxides: Effects of conduction, convection, and internal radiation, J. Crystal Growth, vol. 121 (1992), pp. 473-494.
[3] Q. Xiao and J.J. Derby, The role of internal radiation and melt convection in Czochralski oxide growth: Deep interfaces, interface inversion, and spiraling,
J. Crystal Growth, vol. 128 (1993), pp. 188-194.
[4] J.I. Martinez-Herrera and J.J. Derby, An analysis of capillary-driven viscous flows during the sintering of ceramic powders, AIChE J., accepted for publication (1994).
[5] J.I. Martinez-Herrera and J.J. Derby, Viscous sintering of spherical particles via finite element analysis, J. Am. Ceram. Soc., submitted (1993).
[6] J.J. Derby, S. Brandon, A.G. Salinger, and Q. Xiao, Large-scale numerical analysis of materials processing systems: High-temperature crystal growth and molten glass flows, Comp. Meth. Appl. Mech. Eng., in press (1993).
[7] A.G. Salinger, Q. Xiao, Y. Zhou, and J.J. Derby, Massively parallel finite element computations of three-dimensional, time-dependent, incompressible flows in materials processing systems, Comp. Meth. Appl. Mech. Eng., submitted
[8] Y. Saad and M.H. Schultz, GMRES: A generalized minimal algorithm for solving nonsymmetric linear systems, SIAM J. Sci. Stat. Comp., vol. 7(3) (1986), pp-856-869.
[9] Q. Xiao, A.G. Salinger, Y. Zhou, and J.J. Derby, Massively Parallel Finite Element Analysis of Large-Scale Crystal Growth Processes: Rotating and Coupled Flows, Comp. Meth. Appl. Mech. Eng., submitted (1994).
[10] G.P. Williams, Baroclinic annulus waves, J. Fluid Mech., vol. 49(3) (1971), pp. 417-449.
[11] E.L. Koschmieder, Convection in a rotating laterally-heated annulus, J. Fluid Mech., vol. 51(4) (1972), pp. 637-656.

Finite Element Method with Laplace Transform for Transient Convective Diffusion Problem

Naotaka OKAMOTO and Hideo SAWAMI

Okayama University of Science, Okayama, Japan.

1. Introduction

A numerical method is presented to analyze a transient convective diffusion problem with a first-order chemical reaction defined on an infinite domain. The present method for transient problems is based on the combined boundary and finite element method[1] using Laplace Transform. For the time integration, an inverse Laplace transform presented by Hosono[2] has been successfully employed.

The finite element method is one of the useful numerical tools to analyze these problems. It is however, difficult to analyze a problem defined on the infinite domain by use of the finite element methods. The boundary element method satisfies unconditionally the infinite boundary condition by assuming that the problem is linear. However, the boundary element method has difficulties in solving problems involving non-homogeneous fields because it is difficult to obtain the fundamental solution explicitly. If we appropriately combine the finite element and the boundary element methods, it is obvious that the method which covers the difficulties of both methods can be obtained. For steady problems the combined method is recognized as successful numerical technique to deal with the infinite domain. This method is also useful in transient problems. In the computation of the transient problem using the boundary element method, the domain discretization should be employed to integrate over the infinite domain. Furthermore, difficulties arise when the integral domain should be expanded according to the expansion of the diffusion domain. The above-mentioned difficulties can be overcome by introducing the Laplace transform. The time dependence of the problem can temporarily be removed by this method.

REFERENCES

[1] N.Okamoto and M.Kawahara, Numerical Analysis of Convection - Diffusion with Chemical Reaction by Combined Finite and Boundary Element Methods Int. J. Numer. Methods Fluids, 13, 468-479 (1991).
[2] T.Hosono, "Numerical Inversion of Laplace Transform" Denkigakkai ronbunshi A, 99, 494-500 (1979) (in Japanese).
[3] N.Okamoto, "Analysis of Convective Diffusion Problem with First-order Chemical Reaction by Boundary Element Method", Int. J. Numer. Methods Fluids, 8, 55-64 (1988).

Neural Network Based Parameter Optimizafion for Nonlinear Finite Element Analyses

Genki YAGAWA, Shinobu YOSI-HMURA and Hiroshi OKUDA

University of Tokyo, Tokyo, Japan

1. INTRODUCTION

Computational mechanics technology has progressed dramatically. The use of high performance computers such as massively parallel computers has enabled us to analyze nonlinear large scale problems in various engineering fields. Generally, most of the nonlinear analyses involve several parameters, which greatly influence both the numerical results and the stability. These parameters, e.g., the penalty parameter for incompressible viscous flow analysis and the time step of transient analysis are to be determined appropriately and dynamically referring to convergence behaviors. However, since nonlinear analysis in each field has its own empirical knowledge for choosing optimal parameters, it is difficult to decide the general optimization rule or the procedure for nonlinear finite element analyses.

The present paper describes a new methodology for optimizing multiple parameters of nonlinear finite element analyses using the hierarchical neural network. To demonstrate the performance of the present procedure, it is applied to the parameter optimization of the augmented Lagrangian method for steady state analysis of incompressible viscous flow and the time step optimization of pseudo time-dependent stress analysis for incompressible inelastic structure.

Modeling Impact Phenomena on Vector and Parallel Supercomputers

K.D. Kimsey

U.S. Army Research Laboratory
Weapons Technology Directorate

1. INTRODUCTION

Computer codes for wave propagation and impact have matured considerably since their initial development some 35 years ago. They now serve as valuable tools in studies of materials and structures

subjected to intense impulsive loading. Numerical simulations of high-velocity impact phenomena in two dimensions have been performed routinely for a number of years. The advent of high-performance vector and parallel computers has made three-dimensional solutions of the governing equations tractable.

Large Scale Finite Element Computations Via Iterations and Vectorizations on Modern Workstation Environment

S. NAKAZAWA and R.W. REICH

The Finite Element Factory
1270 Oakmead Parkway, Suite 201
Sunnyvale, California 94086 U.S.A.

1. INTRODUCTION
In this paper, we shall focus on the efficient use of modern computing equipment for high-performance finite element computations. We devote our attention to the evaluation of computational strategies, data structure and flow, algorithmic design and programming techniques. In particular, we investigate the effective use of networked workstations equipped with high-performance microprocessors. Such systems often outperform multi-user supercomputing facilities in terms of the elapsed time to complete certain computationally intensive tasks. Our discussion will be restricted to the implicit numerical processes involving the formulation and factorization of finite element coefficient arrays.

The finite element method has traditionally been deployed on the most powerful equipment because of its computational intensity. Progress in hardware and software technologies has made it possible to tackle complicated engineering and scientific problems on powerful desktop systems. Problems only tractable on supercomputers in the recent past are now routinely solved using these systems.

The use of massively parallel systems are presently limited to the research community. Many architecture and programming paradigms are being proposed and evaluated. Considerable time will be required for such systems to be commonly accepted by engineering community and used for practical applications. Coarse grain parallelization such as symmetric multi-processor systems are emerging on workstations and servers. We would focus our future investigations on the effective use of such systems as a natural extension to what is presented in this paper.

Fundamentals on Infinite-Domain Interpolations

Masayuki OKABE

Optopian Inc., Saitama, Japan

1. INTRODUCTION

The conventional finite element method based on the piecewise polynomial interpolations has limited capability to analyze the phenomena in the wholeor half-space. Here infinite-domain interpolations which can be conformably connected with the isoparametric elements are of great significance.

In such infinite element. our trial function is to be selected over each (semi-)infinite domain so that the dominant decaying mode is reproducible, at least approximately, in the global system. Some infinite elements adopt the special non-polynomial trial function space under the usual polynomial mapping [13, while the others depend on the specific mapping called *the infinity mapping* with polynomial interpolations [23, [3].

This paper is devoted to establish the unified theory in the concerning infinite elementology.

REFERENCES

[1] P. Bettess, Infinite elements, Int. J. Numer. leths. Engrg., vol.11(1977), pp. 53-64.
[2] D.K. Gartling and E.B. Becker, Finite element analysis of viscous, incompressible flow-Part 1: basic methodology, Comput. Yeths. Appl. Mech. Engrg., vol.8(1976), pp. 51-60.

[3] M. Okabe, Generalized Lagrange family for the cube with special reference to infinite-domain interpolations, Comput. Meths.. Appl. Mech Engrg., vo1.38 (1983), pp. 153-168.

Moving boundary simulation for broken dam problem by finite element method

Tsuyoshi UMETSU

Maebashi City College of Technology, Gumma, JAPAN

1. INTRODUCTION

This paper presents a computational technique of the moving water domain based on shallow water equations by two-step explicit finite element method. It is important to know the speed of water up on the drying bed, when the dam *is* broken. The water boundary is calculated by moving boundary technique based on finite element method. In this study, the moving boundary technique is used by finding if an element has water depth or not.

It would be better to use the same technique to find the broken dam problem as in the flood flow river. However, some declinations occurred using such technique, it is convenient to operate with such method. These declinations are present from the simulation of the highest points of the water after the dam *is* broken, *as* if these points are part of the flood flow river. In this case, the velocity of the boundary nodes *is* zero. But in the broken dam problem, the velocity of these points is not equal zero.

In this paper, this problem *is* treated and new technique is introduced to reduce such abnormal phenomenon. The numerical results of a simple broken dam example, are found by old and new' moving boundary technique.

Simulations of Coupled Viscous and Porous Flow Problems [1]

D.K. Gartling, C.E. Hickox and R.C. Givler

Computational Fluid Dynamics Department Sandia National Laboratories
Albuquerque, New Mexico 87185

1. INTRODUCTION

Conjugate problems involving a viscous flow and an adjoining fluid saturated porous medium represent an important class of problems for a variety of technologies. In some applications the interface between the porous material and the clear fluid is distinct and its location is known *a priori*. Typical of these applications are packed beds for thermal or chemical processing, insulating structures, filters and porous bearings. A more difficult problem involves a free boundary interface, where the location of the clear fluid/porous boundary is determined by the variation of one or more dependent variables and therefore evolves with the dynamics of the flow. The prototypical application for this type of conjugate problem is found in the solidification of metal alloys where the evolving dendrite structure is represented by a porous medium. The most general situation for all applications is one in which the interface between the bulk fluid and the porous region is completely open, allowing the free transport of fluid, heat, *etc.* between the regions.

The formulation of this type of conjugate flow problem requires that two inter-related questions be resolved. The equations describing the flow of the viscous fluid, the NavierStokes equations, are well-established and are not debated for this application. The appropriate description for flow in the porous medium is, however, not as evident, nor is the related issue of appropriate boundary and interface conditions clearly understood. Though Darcy's law is a well known and justifiable description for saturated porous flows, it has been criticized for its range of applicability and the limited types of boundary and interface conditions that may be reliably employed with the model. Non-Darcy descriptions, such as the Brinkman equation, seek to remedy the boundary and interface condition problems through the addition of higher order terms to the flow equation. Though this approach eases the boundary condition problem, additional material parameters are introduced that are difficult to quantify. The problem is therefore to

24

select an appropriate equation for the porous flow region in conjunction with the proper interface conditions to couple it to the bulk flow. An additional concern involves the appropriate method of transition from a viscous to a porous flow when the interface is not distinct and is described by a porosity variation.

The objective of the present work was to develop a general method for solving conjugate fluid and porous flow problems using a finite element method. In pursuing this goal several combinations of porous flow equations and interface conditions were examined, compared and evaluated. The following section presents the general mathematical description of the equations used in the study; a following section outlines a number of non-Darcy effects and possible interface conditions that have been evaluated.

An Improved *k-ε* Turbulence Model on High-Speed GSMAC-FEM for Estimation of Wind Load

Shinji KAWAMOTO

Flat Glass Manufacturing & Technology Dept, Nippon Sheet Glass Co., Ltd.,
Anesaki-kaigan 6, Ichihara-city, Chiba, 299-01 Japan

1. INTRODUCTION

The pressure-strain correlation term in the differential stress model is the most important term for accurate estimation of wind load on buildings[1]. The larger the turbulent kinetic energy on the inflow boundary becomes, the more significant the error caused by the imperfection of this term becomes. Moreover the cost-efficiency of the differential stress model is inferior to the k–ε turbulence model. Therefore most of the practical applications of turbulence model in wind engineering field are executed by the k–ε turbulence model.

This paper presents an improved k–ε turbulence model for accurate estimation of wind load on bluff body. The deceleration effect through impingement against wall and adverse pressure gradient, and the acceleration effect through favorable pressure gradient are introduced to the turbulent kinetic energy production term of the conventional standard k–ε turbulence model. The flows around square prism in turbulent streams are computed for validation of the improved model, and the pressure distribution on the surface is improved dramatically compared with the standard k–ε turbulence model.

REFERENCES

[1] S.Kawamoto S .Kawabata   and T.Tanahashi, J. Wind Engrg. Indus. Aero., 46&47(1993), 115-120.
[2] S.Murakami, J. Wind Engrg. Indus. Aero., 46&47(1993), 21-36.
[3] Y.Nagano, M. Tagawa and S. Kondoh, Papers of 23th Symp. on Turbulence(1991), 8-12 (in Japanese).
[4] R.F. Blackwelder and L.S.G. Kovasznay, J. Fluid Mech., vol.53, part 1(1972), 61-83.
[5] S.Kawamoto   and T.Tanahashi, 2nd J-SU CFD Symp., 13(1990), 204-216.
[6] N.Takemitau,Trans. JSME,vol.57, No.540, 2516-2522 (in Japanese).
[7] B.E.Lee,J.Fluid Mech., vol.69, part 2(1975), 263-282.

A numerical study on vortex shedding around a heated /
cooled circular cylinder by three-step Taylor-Galerkin
method

Katsumori HATANAKA

Department of Civil Engineering, Nihon University, Chiba, Japan

1. INTRODUCTION

The vortex shedding around circular cylinder has attracted much attention from researchers who work on both experimental and numerical fluid dynamics. The vortex shedding in the mixed convection phenomenon is more complex due to the temperature induced buoyancy effect and is much more interesting than an ordinary vortex shedding problem. (See [1] or [2], for example.)

In this paper the vortex shedding around a heated / cooled circular cylinder has been investigated numerically by the three-step Taylor-Galerkin / velocity correction method [3]. The attention of the present paper is directed to the change of the vortex shedding patterns under influence of buoyancy forces at a constant Reynolds number. The flow problem under consideration has been successfully simulated by the present finite element scheme.

References

[1] K.S.Chang and J.Y.Sa; The Effect of Buoyancy on Vortex Shedding in the Near Wake of A Circular Cylinder, J.Fluid Mech., vol. 220 (1990), pp.253-266.

[2] K.Noto and R Matsumoto; Generation and Suppression of the Karman Vortex Street upon Controling Surface Temperature of a Cylinder, Numerical Methods in Laminar and Turbulent Flow, vol. 7, Part 1 (1991), pp. 671-984.

[3] C.B.Jiang, K.Hatanaka, M.Kawahara and K.Kashiyama;A Three-Step Finite Element Method for Convection Dominated Incompressible Flow, Computational Fluid Dynamics Journal, vol.1 No.4 (1993), pp.447-466.

[4] K.Hatanaka and M.Kawahara; A Fractional Step Finite Element Method for Convective-Conductive Heat Transfer, Int. J.Num. Methods Heat Fluid Flow, vol.1 (1991), pp.77-94.

[5] J.M.Leone JR; Open Boundary Condition Symposium Benchmark Solution: Stratified Flow Over A Backward Facing Step, Int.J.Num. Meth. Fluids, vol.11 (1990), pp.969-984.

[6] G.Evans and S.Paolucci; The Thermoconvective Instability of Plane Poiseuille Flow Heated From Below: A Proposed Benchmark Solution for Open Boundary Flows, Int.J.Num. Meth. Fluids, vol. il (1990), pp.1001-1013.

Applications of CFD to Wind Engineering

Ran WEI, Akiyoshi SEKINE and Masayuki SHIMURA

Technical Research Institute of MAEDA CORPORATION,Tokyo,Japan

1.INTRODUCTION

We are preparing four topics based on our recent researches for the symposium. Those are the influence of open boundary width on numerical results of pressure coefficients, response characteristics of a vortex-exited oscillation of cylinder in low Reynolds numbers flow, response characteristics of a vortex-exited oscillation of cylinder in high Reynolds numbers flow and a term wise comparison between FDMs and FEMs by using Navier-Stokes equation. In this paper, a brief description on the four topics is described.

Analyses Of Mold Flow And Residual Stress In Resin Transfer
Molding Using The Homogenization Method

Whie Chang and Noboru Kikuchi
The University of Michigan, Ann Arbor, Michigan 48109

Resin Transfer Molding

Resin transfer Molding (RTM) is a relatively new process for manufacturing fiber reinforced polymer composites. In RTM, fiber mats which act as reinforcements are cut into the desired shape and arranged inside the mold cavity. Then the mold is closed and the resin is injected through the injection ports of the mold progressively impregnating the fiber mats. When the mold is completely filled, the resin solidifies in the curing phase in which chemical reaction and significant temperature variation take place. After the curing the product is taken out of the mold.

The process analysis is a very important step in the product development stages of RTM. It consists of the mold flow analysis of the mold filling phase and the residual stress analysis in the curing phase. The former can provide useful information to a mold designer such as pressure distributions and flow front profiles within the mold cavity which can be used to predict operating parameters, possible location of defects, hard to fill regions, and proper locations of air tabs and injection ports. The latter can be used to predict the residual stresses which are, in some cases, large enough to cause micro cracking and exhaust the strength of the material. Conventionally, these data were obtained based on experience and repeated experiments which are time consuming and inefficient. In order to optimize the manufacturing process and reduce the cost, numerical analysis of the RTM process is necessary.

In this work, a complete analyses are given which encompasses the formulations and the numerical implementations of the mold flow and of the residual stress. The mold flow analysis deals with the resin flow through the complex pore structures of the reinforcing fiber perform, and the residual stress analysis deals with the stress distribution in the micro structure of the composite. In both analyses, the homogenization method is utilized extensively due to the complexity of the geometry.

The homogenization theory deals with the partial differential equations of physics in heterogeneous materials with periodic structures when the characteristic length of the period is small. In the homogenization theory, it is assumed that the composite is locally formed by spatial repetition of a "microscopic cell". Using this assumption, an equivalent (or "homogenized") material model is found by solving the behavior within the microscopic cell. The theory appeared in the 1970's and has been the subject of considerable research in different areas of applied mechanics. The application of the theory to the analyses of composite materials can be found in Sanchez-Palencia [4] and some computational works can be found in Guedes[8] and Cheng[3]. The fundamentals of the theory can be found in the works of Lions[7], Benssousan, Lions, and Papanicoulau [1], and Oleinik[9].

REFERENCES

[1] A. Benssousan, J.L. Lions, and G. Papanicoulau, *Asymptotic analysis for periodic structures*, North Holland, Amsterdam, 1978

[2] B.D. Harper and Y. Weitsman, *Residual thermal stresses in an unsymmetric crossply graphite epoxy*, Proceedings of the AIAA/ASME/ASCE/AHS 22nd SSDM conference, American Institute of Aero and Astro., 1981, p.931

[3] C.-H. Cheng, *Modeling of the thermal elasto-plastic behavior for composite materials using the homogenization*, 1992, Ph.D. dissertation thesis, Dept. of Mechanical Engineering and Applied Mechanics, The University of Michigan.

[4] E. Sanchez-Palencia, *Homogenization techniques for composite media*, Lecture notes in physics 272, Springer-Verlag, Bedin Heidelberg, 1987

[5] H. Lee and H. Sin, *Iterative boundary pressure reflection method for the simulation of injection mold filling*, Polymer Eng. and Sci., 1990, Vol.30, No. 23, pp. 15131522

[6] H.T. Hahn and N.J. Pagano, *Curing stresses in composite laminates,* Journal of composite materials, Vol. 9, Jan. 1975, pp. 91-106

[7] J.L. Lions, *Some Methods in the Mathematical Analysis of Systems and Their Control,* Science Press, Beijing, China 1981, Gordon and Breach, New York

[8] J.M. Guedes, *Nonlinear computational models for composite materials using homogenization,* Ph.D. dissertation, Dept. of mechanical eng. and applied mechanics, The University of Michigan, 1989

[9] O. Oleinik, *On homogenization problems,* In: Ciarlet, P.G. and Rouseau, M. (ed), Trends and applications of pure mathematics to mechanics, 1984, Berlin: Springer Verlag.

[10]T. J. Chapman, J.W. Gillespie, Jr. and R.B. Pipes, *Prediction of process-induced residual stresses in thermoplastic composites, J. of composite materials,* Vol. 24, Jun. 1990, pp. 616-643

[11 IV. W. Wang, C.A.Hieber and K. K Wang, *Dynamic simulation and graphics for the injection molding of three-dimensional thin parts, J.* of Polymer Eng., Vol. 7, 1986, pp. 21-45.

[12]Y. Weirsman, *Residual thermal stresses due to cool-down of epoxy-resin composites,* Journal of applied mechanics, Vol. 46, Sep. 1979, pp. 563-567

[13]Z. Tadmor, E. Broyer, and C. Gutfinger, *Flow Analysis Network (F.4N)--A method for solving flow problems in polymer processing,* Polymer Engineering and Science, **Vol.** 14, 1974, pp. 660-665.

An Adaptive Mesh Rezoning for FSI Problems

Yoshihiro TAKI

Meijo University, Nagoya, Japan

1. INTRODUCTION

It is quite useful to develop an adaptive mesh rezoning technique which adjusts to the moving domain and update the mesh, yet, maintaining the element connectivity, since the Space-time Finite Element Method is considered as a promising framework for fluid-structure interaction (FSI) problems. We have developed the adaptive mesh rezoning technique with the least-square constrained conditions, which save the element, locating close to the moving boundary, from breakdown trouble and keep well-conditioned shape. The our technique is entirely general in that it can be applied to structured and unstructured meshes, effectively.

2
REFERENCES

[1]. A. Masud and T.J.R. Hughes, An Adaptive Mesh Rezoning Strategy for FluidStructure Interaction, Research Brief, Division of Applied Mechanics, Stanford University, January, (1991).

Finite Element Computations of Periodic Adsorption Processes

George M. HARRIOH'

Air Products and Chemicals, Inc. 7201 Hamilton Boulevard Allentown, PA 18195-1501 USA

Adsorption, in which molecular constituents of a fluid are partitioned by virtue of having different affinities for a solid, is used to separate bulk mixtures as well as remove trace impurities. Adsorption processes are typically carried out in large columns packed with a multitude of small porous pellets to promote contact between fluid and solid. Species in the feed stream are fractionated by the solid on an 'adsorption' step which is then followed by a sequence of 'rinse' or 'purge' steps to clean out the solid so that it may be used again. The dynamics vary in detail throughout the column, but at each point eventually a cyclic or periodic state is reached in which the particles undergo equal amounts of adsorption and desorption.

[1] G.M. Harriott, Memory-Integral Mass-Transfer Models for Adsorption Process Simulation, AICkE J., 39, 422 (1993).

[2] E. Glueckauf, Theory of Chromatography Pan 10-Formulae for Diffusion Into Spheres and Their Application to Chromatography, Trans. Farad. Soc., 51, 1540 (1955).

[3] M.D. LeVan and D.T. Croft, Determination of Periodic States of Pressure Swing Adsorption Cycles, presented at: AIChE Annual Meeting, Los Angeles, November 1991.

A One-Point Quadrature Technique with a New Hourglass Controller for Large Scale Finite Element Fluid Analysis

Hiroshi OKUDA and Genki YAGAWA

University of Tokyo, Tokyo, Japan

1. INTRODUCTION

A large storage requirement of the finite element method, which is a result of using unstructured meshes and the need to solve the system of equations. The one-point quadrature technique proposed by Gresho et al.[1] is known to be quite efficient in saving computational storage. By assuming the one-point quadrature, the diffusion and the advection matrices become storage-free since they can be computed from the gradient Matrix. Some numerical dumping needs to be introduced to suppress physically meaningless oscillations called the 'hourglass mode', which might be caused by the reduced (one-point) integration of the diffusion matrix. The authors have developed an hourglass controller[2], which can incorporate the effect of an aspect ratio of element. The authors also have proposed a matrix-storage free formulation. Since the gradient matrix still occupies large amount of memory, by virtue of developing an efficient algorithm to compute the gradient matrix, a matrix-storage free formulation has been made possible. Comparably practical problems having several tens of thousands of nodes can be easily handled by currently available engineering workstations.

REFERENCES

[1] P.M. Gresho, S.T. Chan, R.L. Lee and C.D. Upson, Int. J. Num. Meth. Fluids, Vol.4, pp.557-598, 1984.
[2] H. Okuda, Y. Eguchi and G. Yagawa, Trans. JSME, Ser. B, Vol.54, No.498, pp.337-344, 1987. (in Japanese)
[3] U. Ghia, K.M. Ghia and C.T. Shin, J. of Compt. Phys., Vol.48, pp.387-411, 1982.

Multi-Resolution Reproducing Kernel Particle Methods

WING KAM LIU and YIJUNG CHEN

Northwestern University, Evanston, IL, USA

1. INTRODUCTION

Multiple scale methods [1-4], which are based on discrete and continuous reproducing kernels, wavelets, and integral window transforms, are developed. In this development, a microscope is constructed with a flexible space/time localized window function which translates and dilates in space/time to cover the entire domain of interest. This microscope can magnify, examine, and record the image of the various scales/frequencies of the response locally within the support of the window function. The degree of magnification will depend on the power of the microscope, a flexible space-scale/time-frequency window function. This complete characterization of the unknown response is performed through the integral window transform. This localization process can be achieved by dilating the flexible multiple-scale window

function. A parameter defined by the energy error ratio is introduced to pin point the scale/frequency domain of interest, and to prevent aliasing from the sampling procedure. The zoom in and zoom out capability of the window function is especially useful in examining complex flow phenomena, such as flow induced vibration, dynamic stability of flow-structure interaction, turbulence structures, and high frequency structural dynamic response. Numerical examples, which include the Helmholtz equation and the advection-diffusion equation, are presented and compared with finite element results.

REFERENCE

[1] Liu, W.K., Chen, Y. and Oberste-Brandenburg, C. "Reproducing Kernel and Wavelet Particle Methods and Accuracy Control," (preprint) (1993).
[2] Liu, W.K. Oberste-Brandenburg, C. "Reproducing Kernel and Wavelet Particle Methods," (preprint) (1993).
[3] Liu, W.K., Jun, S., Adee, J. and Belytschko, T., "Reproducing Kernel Particle Methods for Structural Dynamics," (preprint) (1993).
[4] Liu, W.K., Zhang, Y., Belytschko, T., Jun, S., Adee, J. and Gu, L., "Reproducing Kernel Methods," International Journal of Numerical Methods in fluids, Submitted May (1993).

An Optimal Control Analysis of Water Pollution Problem

Hirokazu HIRANO * and Kenichi YOSHIDA **

*Faculty of Policy Studies, Chuo University, Tokyo, Japan
**Department of Civil Engineering, Chuo University, Tokyo, Japan

1. INTRODUCTION

In general, water quality control is one of the most important social problems especially closed water region. If we want control this water pollution problem, we have to consider the inequality constrains. Until now, we have been studying about the dynamic optimization without constrain problem for first step. This type problem can be solved *by* Conjugate Gradient *(CG)* Method, Dynamic Programming (DP) Method, etc.. However these methods cannot be solved under inequality constrain. Therefore this paper proposed a method based the Sakawa-Shindo method to solve the water pollution control problem with inequality constraints on functions of control.

In this study, the optimal control analysis has been applied to water pollution problem. The main purpose of control of water pollution problem is to reduce the contaminated water, i.e. the determination of control values to keep the water quality which satisfy a criterion of concentration at arbitary point in the flow field.

In the present optimal control analysis, the linear shallow water equation and the convection diffusion equation are used as the governing equations for flow field. The explicit Euler method is applied to discretize the shallow water equation and the convection diffusion equation. The simulations of simple model and applications to the water pollution problems have been carried out. The results of the method are quite resortable. The application for practical problem will be carried out.

Finite Element Heterogeneous Algorithms for Transient Aeroelastic Computations

C. FARHAT [1]

[1]University of Colorado, Boulder, CO, USA

1. A THREE-FIELD FORMULATION OF COUPLED AEROELASTIC PROBLEMS

In order to predict the dynamic response of a rigid or flexible structure in a fluid flow, the equations of motion of the structure and the fluid must be solved simultaneously. The most difficult part of handling numerically the fluid/structure coupling stems from the fact that the structural equations are usually formulated with material (Lagrangian) co-ordinates, while the fluid equations are typically written using

spatial (Eulerian) co-ordinates. Therefore, a straightforward approach to the solution of the coupled fluid/structure dynamic equations requires moving at each time step at least the portions of the fluid grid that are close to the moving structure. This can be appropriate for small displacements of the structure but may lead to severe grid distortions when the structure undergoes large motion. Several different approaches have emerged as an alternative to partial re-gridding in transient aeroelastic computations, among which we note the Arbitrary Lagrangian Eulerian (ALE) formulation [1], the co-rotational approach [2,3], and dynamic meshes [4]. All of these approaches treat a computational aeroelastic problem as a two-field problem.

REFERENCES

1. J. Donea, An arbitrary Lagrangian-Eulerian finite element method for transient fluid-structure interactions, Comput. Meths. Appl. Mech. Engrg., vol. 33 (1982), pp. 689-723.
2. O.A. Kandil and H.A. Chuang, Unsteady vortex-dominated flows around maneuvering wings over a wide range of reach numbers, AIAA Paper No. 880317, AIAA 26th Aerospace Sciences Meeting, Reno, Nevada, January 11-14, 1988.

3. C. Farhat and T.Y. Lin, Transient aeroelastic computations using multiple moving frames of reference, AIAA Paper No. 90-3053, AIAA 8th Applied Aerodynamics Conference, Portland, Oregon, August 20-22, 1990.
4. J.T. Batina, Unsteady Euler airfoil solutions using unstructured dynamic meshes, AIAA Paper No. 89-0115, AIAA 27th Aerospace Sciences Meeting, Reno, Nevada, January 9-12, 1989.
5. M. Lesoinne and C. Farhat, Stability analysis of dynamic meshes for transient aeroelastic computations, AIAA Paper 93-3325, 11th AIAA Computational Fluid Dynamics Conference, Orlando, Florida, July 6-9, 1993.
6. C. Farhat, L. Fezoui, and S. Lanteri, Two-Dimensional viscous flow computations on the Connection Machine: unstructured meshes, upwind schemes, and massively parallel computations, Comput. Meths. Appl. Mech. Engrg., vol. 102 (1993), pp. 61-88.
7. C. Farhat and F.X. Roux, A method of finite element tearing and interconnecting and its parallel solution algorithm, Internat. J. Numer. Meths. Engrg., vol. 32 (1991), pp. 1205-1227.
8. C. Farhat, H.D. Simon and S. Lanteri, TOP/DOMDEC, A software tool for mesh partitioning and parallel processing, (in press).
9. N. Marman and C. Farhat, Matching fluid and structure meshes for aerodastic computations: a parallel approach, (in press).

Estimation of Wave Propagation using Kalman Filter

Toshimitsu TAKAGP[1], Kosuke INAMOTO[2], and Mutsuto KAWAHARA[2]

1. INA Corporation, Tokyo, Japan
2. Chuo University, Tokyo, Japan

1. INTRODUCTION

In the coastal management for the coastal defence against erosion, storm surge, etc, and for the use as hatbouts and resort areas, it is very important to estimate or predict the currents such as tidal or near shore currents, wave propagation, tidal elevation and topography change. However, for the reason that these phenomena are very complicated and stochastic, it is difficult to estimate them determinately. On the other hand, sometimes they are observed or monitored using various kinds of equipment. Those data often includes erroneous disturbance. In this paper, the Kalman Filtering techniqe is applied to the wave propagation analysis, consequently it is possible to take into account the mesurement data efficiently to get much information about the wave movement.

31

Finite Element Approximations to
Axisymmetric Flow Problems

Masahisa TABATA
University of Electro-Communications, Chofu, Tokyo, Japan

1. INTRODUCTION

We consider finite element approximations to axisymmetric flow problems. The differential operators in these problems have coefficients of negative powers of the distance from the axis. So we have to treat weighted Sobolev spaces [1]. The radius and the axis components of the velocity belong to different function spaces. So we have to deal with three weighted Sobolev spaces for the velocity and the pressure.

As the model problem we consider an axisymmetric stationary Stokes problem. We can apply the theory of the mixed finite element method [2] and obtain the convergence under the inf-sup condition (Babuska-Brezzi condition) similarly to the case in the Cartesian coordinates. Recently Franca et al. [3],[4] have developed the stabilized finite element method where the inf-sup condition is not required Any combination of velocity approximating space and pressure approximating space is possible. We show that the stabilized method can also be applied to the axisymmetric problem under a suitable subdivision of the domain and a proper choice of base functions. An error estimate in the proper norm is also obtained.

References

[1] Mercier, B. and Raugel, G., "Resolution d'un probleme aux limites dans un ouvert axisymetrique par element finits en r, z et serie de Fourier en 0", RAIRO Analyse Numerique/Numerical Analysis, 4(1982), 405-461.

[2] Brezzi, F., "On the existence, uniqueness and approximation of saddle point problems arising from Lagrangian multipliers", RAIRO Analyse Numerique/Numerical Analysis, 8-a2(1974), 129-151.

[3] Franca, L.P., Frey, S.L., "Stabilized finite element methods: II. The incompressible Navier-Stokes equations", Computer Methods in Applied Mechanics and Engineering, 99(1992), 209-233.

[4] Franca, L.P. and Stenberg, R., "Error analysis of some Galerkin least squares methods for the elasticity equations", SIAM J. Numer. Analysis, 28(1991), 1680-1697.

[5] Tabata, M., "Stabilized finite element approximation to the axisymmetric Stokes problem", to appear.

NUMERICAL SIMULATION OF FREE BOUNDARY PROBLEMS

S. Chippada[1], B.Ramaswamy[2] and S.W. Joo[3]

Introduction

Boundary/Initial value problems in which part or whole of the flow domain is not known apriori and needs to be determined as part of the solution procedure are called free boundary problems. Problems of this nature are of interest in crystal growth processes, open channel flows, melting and solidification, jets, flame propagation, groundwater seepage, metal forming processes, draining film flows, and various other fields of engineering. Additional non-linearity is introduced into an already non-linear governing equations due to the unknown free boundary position, Due to this reason, analytical treatment of these problems is extremely difficult. In this paper a numerical model developed based on the Arbitrary Lagrangian Eulerian(ALE) description of flow is described. This procedure has been used to solve open channel flow, thermocapillary and buoyancy driven flow, and draining film instabilities, all in two dimensions.

[1]Grad. Student, Mech. Engg., Rice Univ.
[2]Asst. Prof., Mech. Engg., Rice Univ.

3 Asst. Prof., Mech. Engg., Wayne State Univ.

Air Flow Computation
around an Automated Guided Vehicle

HirosM KANAYAMA[1], Kohji TOSHIGAMI[1], Yukari TASHIRO[1],
Masahisa TABATA[2] and Shoichi FUJIMA[2]

1 FUJIFACOM Corporation, Kanagawa, Japan
2 The University of Electro-Communications, Tokyo, Japan

1. INTRODUCTION

An upwind finite element scheme for the incompressible viscous flow at high Reynolds number was proposed by Tabata and Pujima[l]. The scheme has the potential to appropriate the advection term in third-order accuracy. We apply it to a two-dimensional non-stationary analysis of air flow around an Automated Guided Vehicle (AGV). The results are at least qualitatively good and compatible with experimental ones.

References

[1] M.Tabata and S.Fujima, An upwind finite element scheme for Mgh-Reynoldsn-tuber flows, Int. J. Numer. Meth. Fluids, vol.12 (1991), pp.305-322.
[2] M.Bercovier and O.Pironneau, Error estimates for finite element method solution of the Stokes problem in the primitive variables, N-met. Math., vol.33 (1979), pp.211-224.

A Visualization of Flows by FEM
using Six-node Triangular Element

YOJI SHIMAZAKI
Tokai University, Kanagawa, Japan

1.   INTRODUCTION
        It is one of the most important areas in technological and engineering fields to calculate transient flows, such as those found in flows of soils, cements, metals, and polymers etc. There are so called particle-in-cell (PIC) method[1] and markerand-ceil(MAC) method[2] analyzing the flows. These methods make use of marker particles to show free surfaces. The flow patterns can be visualized if the markers are also arranged inside the transient flow. These methods, however, are difficult to apply under complicated fixed boundary conditions. Such conditions can easily be covered if the finite element method is used. Using the finite element method, markers can be also used for the analysis. Shiojima et al. have introduced the area co-ordinate system of the linear triangular element, which is utilized for finding the new marker position, into a four-node rectangular isoparametric element. The two kinds of interpolation systems, however, give complicated solution algorithm and may give unexpected errors on calculated results. Also, this method requires liner mesh arrangement to obtain an accurate result. This implies that a large amount of extra storage may be required for the analysis.
in this study, we present a finite element method using six-node triangular isoparametric element for the analysis, in which marker particles are introduced to represent the transient creeping fluid flow motion. For determining the marker positions in an element, area co-ordinate system of the triangular element is used. With the six-node element, the determination of the new marker position in an element becomes very simple compared with those using the linear triangular element. It is because that we can make use of three mid-node points in each side of the triangle to the corresponding three area co-ordinates. Also the velocities at the six nodes obtained by the calculation can be directly used for the interpolation of the marker velocity. The program becomes simple and higher order accuracy can be expected, and course mesh arrangement can be available with the present method. To verify the scheme, we present a simple axisymmetric problem. The time dependent creeping flow of incompressible Newtonjan fluid is considered in this study..

IDENTIFICATION OF THERMAL CONDUCTIVITY OF GROUND MATERIALS
BY NONLINEAR LEAST SQUARE METHOD

Keiichi KOJIMA[1], Noriyoshi KANEKO[1], Toshio KODAMA[1],
and Mutsuto KAWAHARA[2]

1 Sato Kogyo Co., Ltd., Tokyo, Japan
2 Chuo University, Tokyo, Japan

1. INTRODUCTION

A nonlinear least square method which numerically identifies the thermal conductivity of ground materials is presented in this paper. The performance function expressed by the sum of squared differences between observed and computed temperature is employed to estimate the thermal conductivity. The Fletcher Reeves method is chosen as the methods to minimize the performance function. The identification analysis of the thermal conductivity of the ground is carried out by using the temperature measured in an actual field.

Explicit Crank-Nicolson Method with Finite Element

Michio SAKAKIHARA
Okayama University of Science, Okayama, Japan

1. INTRODUCTION

The Crank-Nicolson(CN) method is a kind of useful numerical time integration methods for simulating some transient physical phenomena such as heat or mass transfer, wave propagation and fluid flow problems. Although the method has some advantages as unconditionally stable and obtaining second order accuracy in time, that has some difficulties to carry out a calculation for some large scale computations since that is a kind of implicit schemes. The author and co-workers presented the block-wise local CrankNicolson(LCN) method to solve one- and two-dimensional heat transfer problems with the finite difference method[1]. The method inherits some advantages from the CN method even if the LCN method is formulated with an explicit form. However the LCN method with block-wise decomposition is not extensible to three-dimensional problems. The aim of the paper is to present an explicit numerical integration, which is unconditionally stable and has the second accuracy in time, for a semi-discretized equation which is derived from applying the finite element method in time and the mass lumping technique. The present method is carried out with explicit calculations, that is, the method is formulated without linear solvers as Gauss-Jordan, LU-decomposition, Gauss-Sidel, SOR method, etc.. Therefore the author call the present method the explicit Crank-Nicolson method. We show the idea of the present method and some properties.

[1] Abudolashit,A., Sakaldhara,M. and Niki, H., Local Crank-Nicolson method for solving heat equation, Hiroshima Math. J., vol.23 (1993) in press.